

Response Spectrum Analysis

Shock and Seismic

FEMAP & NX Nastran



Table of Contents

1.	INTRODUCTION.....	3
2.	THE ACCELEROGRAM.....	4
3.	CREATING A RESPONSE SPECTRUM	5
4.	NX NASTRAN METHOD.....	8
5.	RESPONSE SPECTRUM ANALYSIS EXAMPLE: CANTILEVER BEAM.....	10
5.1	PROBLEM DEFINITION.....	11
5.2	ANALYTICAL SOLUTION.....	12
5.3	STEP 1: CREATING THE FE MODEL.....	15
5.4	STEP 2: DEFINING THE RESPONSE SPECTRUM.....	16
5.5	STEP 3: CREATING INTERPOLATION TABLE.....	17
5.6	STEP 4: CREATING A MODAL DAMPING FUNCTION	18
5.7	STEP 5: CREATING THE EXCITATION NODE.....	19
5.8	STEP 6: CONSTRAINING THE MODEL.....	20
5.9	STEP 7: SETTING UP THE ANALYSIS.....	21
5.10	POST PROCESSING THE RESULTS	28

1. INTRODUCTION

Response spectrum analysis is widely used for the design and assessment of structures that are subject to earthquakes or shock events. The reason we want to use a response spectrum is that it allows us to analyze transient events (time based events) without having to review hundreds or thousands of results sets. In essence, it allows us to assess the maximum dynamic response (stress, acceleration, velocity or displacement) of a structure using a very simple analysis technique (normal modes). Moreover all of this goodness can be had by only having to interrogate one (1) output set.

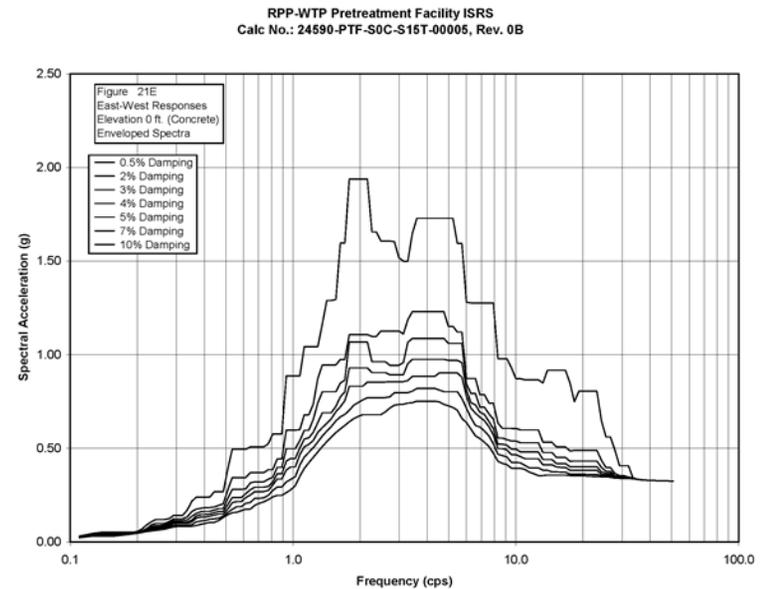
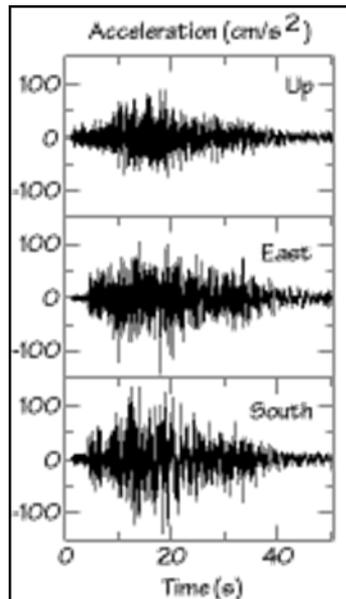
This tutorial will walk you through the theoretical background of response spectrum analysis and how to actually implement it within FEMAP & NX Nastran.



2. THE ACCELEROGRAM

The whole key to a response spectrum analysis is generating the response spectrum that is used to drive the FEA model. A response spectrum is the “load” to the FEA model that allows us to reduce a complicated transient analysis (time based) to simple normal modes analysis (frequency based). In general terms, response spectrums are generated from acceleration versus time measurements or accelerograms. An example of an accelerogram is shown below for an earthquake event. The three traces (Up, East and South) were generated by a tri-axial accelerometer mounted onto a concrete foundation. Similar traces can be obtained during shock events (e.g., rocket launches).

A response spectrum (an example is shown on the right) is created in a somewhat non-intuitive manner from an accelerogram. The conversion from time based data into a response spectrum is mathematically complex (Note: it is not a Fast Fourier Transform (FFT)). In the next section we will give an equation-less explanation of this mathematical procedure.

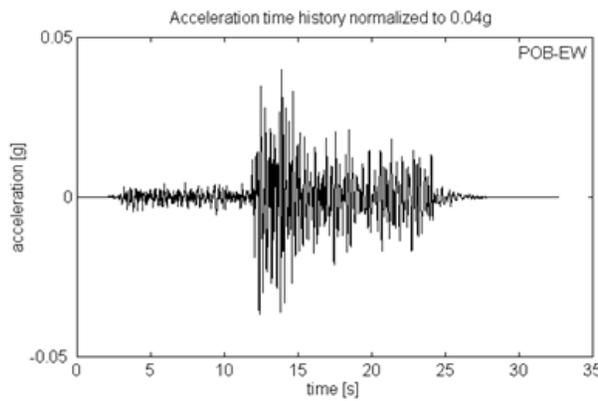
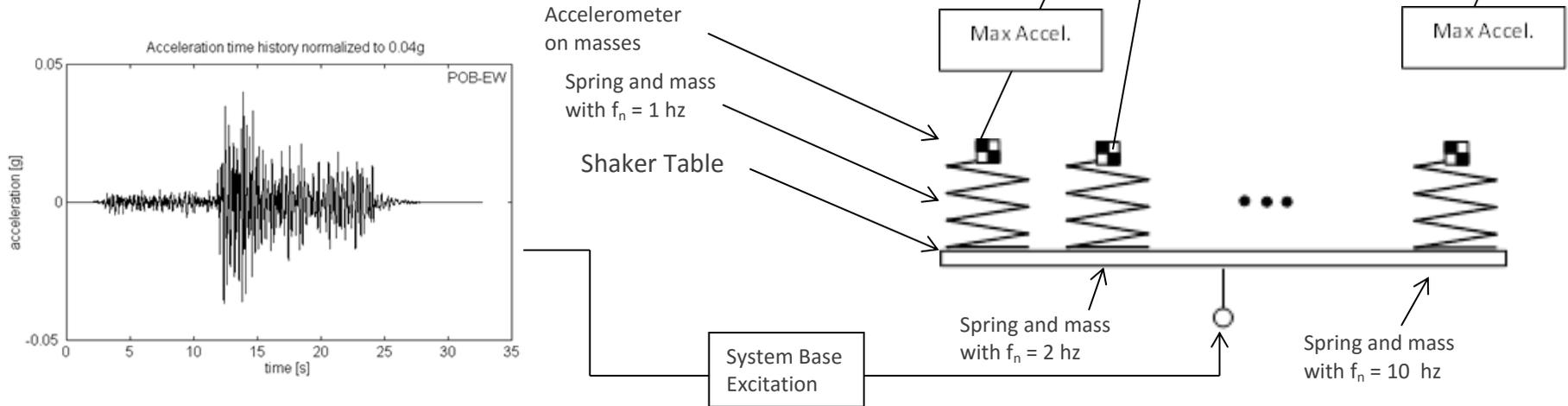
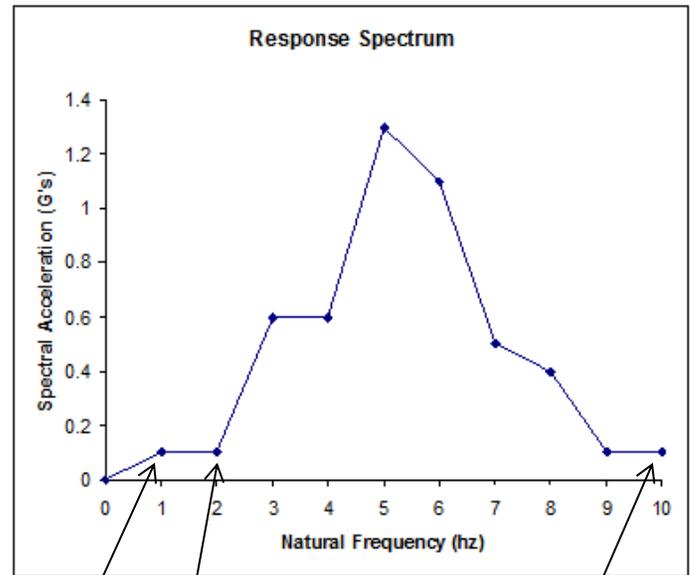


Accelerogram image courtesy of Charles Ammon, Penn State), <http://earthquake.usgs.gov/learning/glossary.php?term=accelerogram>

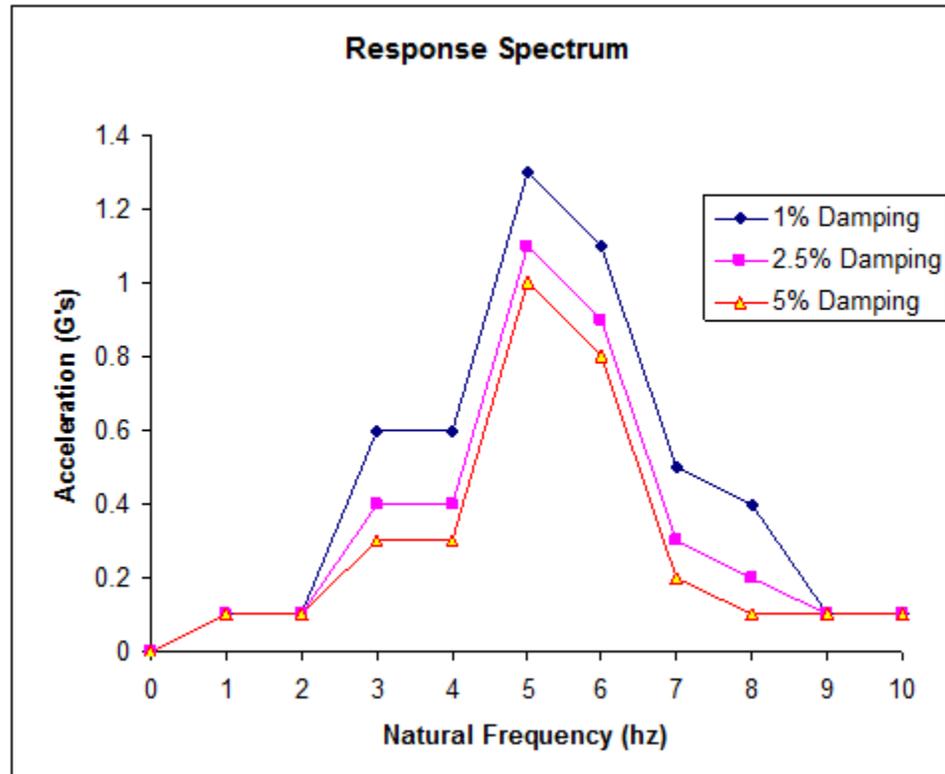
The response spectrum image is courtesy of the U.S. Taxpayers.

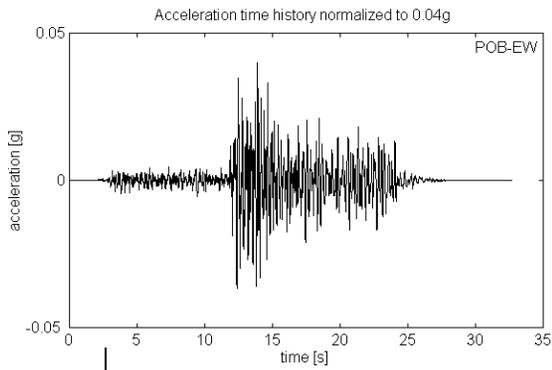
3. CREATING A RESPONSE SPECTRUM

As we mentioned before, converting an accelerogram (Time domain) to a response spectrum (frequency domain) is not done with a FFT. Let's explore this procedure using a simple example. We want to create a response spectrum that goes from 0 to 10 Hz using our acceleration versus time history data (i.e., accelerogram). Think of a shaker table having ten little spring/mass units attached onto its surface. At the top of each spring is a mass block. To measure the acceleration of the spring/mass system we have fixed an accelerometer onto the block. Our spring/mass systems are then tuned to range in eigenvalues or natural modes from 1 to 10 Hz in 1 Hz increments. The shaker table is then driven with our accelerogram. The output from the ten accelerometers are normalized with respect to gravity giving us g's (dimensionless). Our final step is to plot these normalized values against their respective eigenvalues. The resulting graph is then called a "Response Spectrum". Of course, there exists a purely mathematical approach for turning accelerograms into response spectrums but the theory is a bit dry and left for graduate students in mechanics.



To account for system damping (which exists to some degree in all structures), we can attach dampers to our springs and rerun the shaker table experiment. As you might expect, another curve is generated that follows the exact same profile but with lower acceleration values. Below is a classic example of a family of response spectrum curves for 1%, 2.5% and 5% of critical damping.



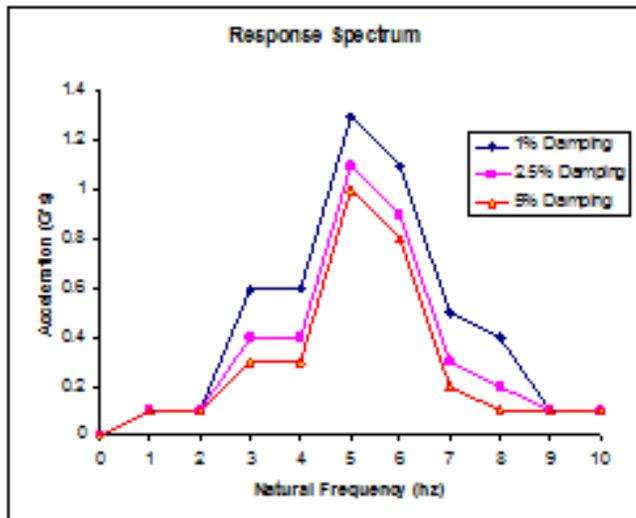


Measure acceleration from an earthquake or shock event (accelerogram)

Use accelerogram to excite a system of springs with unique natural frequencies

Plot maximum acceleration of each individual spring against their natural frequency

Repeat with different levels of oscillator damping



4. NX NASTRAN METHOD

Nastran starts off the response spectrum analysis by calculating the normal modes of the system and then the modal participation factors (PF). We seek the solution to;

$$([K] - \lambda [M])[Ø] = 0, \lambda = \omega^2$$

Which is an eigenvalue problem, the non-trivial solution to the problem is found from;

$$\det([K] - \lambda [M]) = 0$$

This gives non-zero eigenvectors $[Ø] \neq 0$. The response spectrum is used in conjunction with the PF's and a modal combination method to determine the system response to the spectrum. The PF's are calculated by;

$$PF = [Ø]^T [M] [I]$$

Where $[Ø]$ is the mode shape vector, M is the mass matrix and I is a unit vector of the same dimensions as M.

What we are saying is that each normal mode shape has a bit of mass associated with it. To generate displacements and stresses within the structure, each mode shape (and its PF) is factored against the response spectrum at that particular frequency. That is to say, a standard linear stress analysis is performed at each normal mode submitted in the response spectrum analysis. After all of the requested mode shapes have been processed, the resulting displacements and stresses are summed up using some sort of combination method (see next slide) and then written out into one final result set for subsequent post-processing.

As we had mentioned, at the end of the analysis, the individual results set generated at each mode are summed up. An idea of some of the methods that are available:

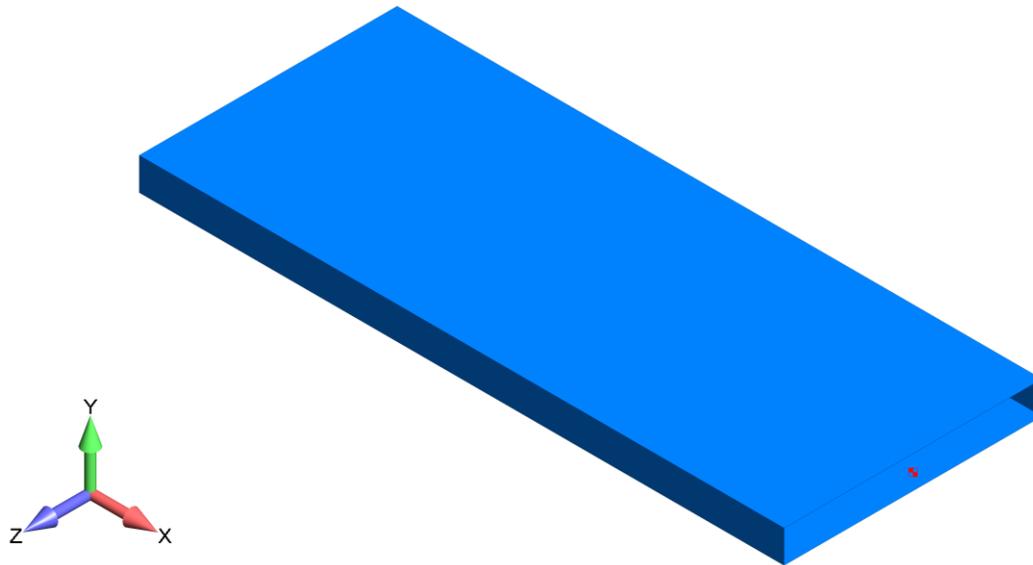
1. ABS – This combination method is the most conservative, the absolute value resulting from each mode are summed.
2. SRSS – Square root of the sum of the squares, also contains a provision to use the ABS method for mode combination that are within a user specified closeness (default is 1.0).
3. NRL – U.S. Navy Shock Design Modal Summation Convention, this method combines the SRSS and the ABS methods
4. NRLO – Updated NRL method to comply with NAVSEA-0908-LP-000-3010 specification.

The choice of the combination method is dependent on the analysis, the SRSS method is generally the method of choice since it includes the provision for using the ABS method for close modes.

5. RESPONSE SPECTRUM ANALYSIS EXAMPLE: CANTILEVER BEAM

Doing a Response Spectrum Analysis in Femap:

1. Creating the FE model
2. Defining the response spectrum or spectrums
3. Creating the interpolation table
4. Creating a modal damping function
5. Creating the excitation node and tying it into the model
6. Constraining the Model
7. Setting up the Analysis in the Analysis Manager
8. Post-Processing the results



5.1 PROBLEM DEFINITION

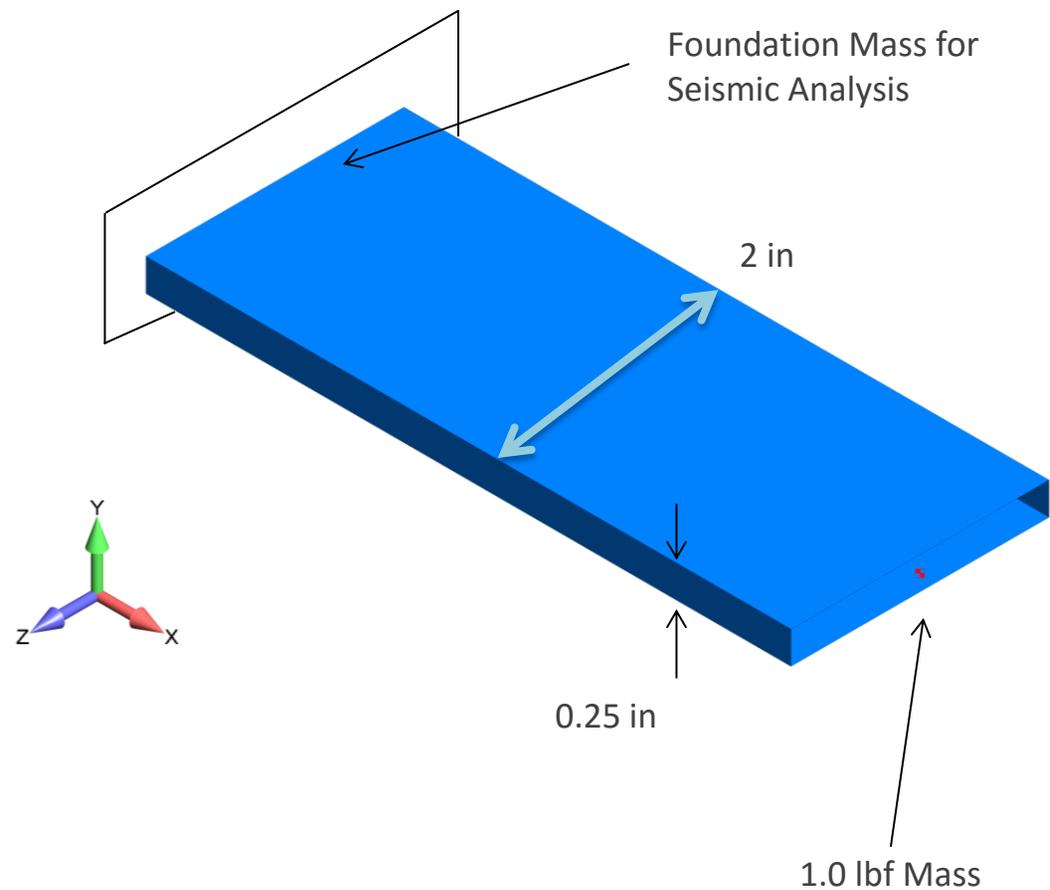
A cantilevered beam six inches in length is used to support a mass of 1.0 [lbs·s²/in]. Our objective is to determine the dynamic stresses of the beam for vibration along the vertical axis.

The FEA model is a single beam element. A picture of the beam element, with its cross section displayed is shown on the right.

We will compare the FEA results to an analytical solution.

Our unit system is lb/in/s and

1 g = 386 in/s². The elastic modulus of steel is taken as 2.9E+07 psi.



5.2 ANALYTICAL SOLUTION

The stiffness of the cantilever beam can be determined from the Young's modulus for steel, the moment of inertia and the length of the beam:

$$k = \frac{3EI}{L^3} = \frac{3 * 2.9E7 * \left(\frac{2 * 0.25^3}{12}\right)}{6^3} = 1048.9 \frac{lbs}{in}$$

With the stiffness and the weight, the circular natural frequency can be determined:

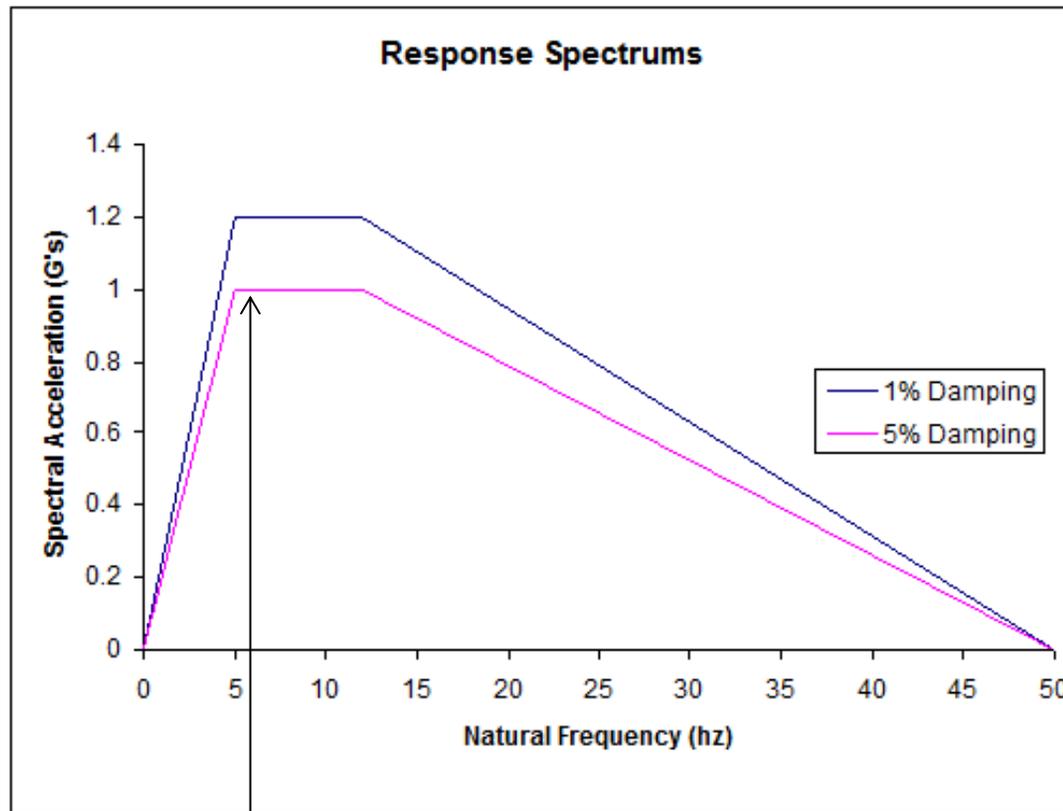
$$\omega_n = \sqrt{\frac{k}{m}} = \sqrt{\frac{1048.9}{1.0}} = 32.387 \frac{rad}{s}$$

And finally the natural frequency can be found:

$$f_n = \frac{\omega_n}{2\pi} = \frac{32.387}{2\pi} = 5.1545 \text{ Hz}$$

We will determine the systems response to a simple response spectrum, to highlight the abilities of Nastran, we will use a damping value of 3%, this value is intermediary between the given response spectrum curves (just like a standard engineering problem, i.e., we never get exactly what we want for an analysis). This is no big deal since NX Nastran can interpolate between the given damping values.

Values for the problem have been chosen in order to provide a clean and straight forward solution. Since the beam model is a 1 degree of freedom system, we know that the modal participation factor (PF) for the above calculated natural frequency will be 1.0 (since all the mass of the system is at the only mode that the system has – never in real life but great for this example).



5.155 Hz

From the value that we previously determined from the natural frequency (5.155 Hz), using the response spectrum shown below, we can see that the response will be 1.2 and 1.0 G's. The system is linear, and as such linear interpolation can be used to determine the system response with 3% damping.

- Response with 1% damping (Sa) – 1.2 G's
- Response with 3% damping (Sa) – 1.1 G's (using linear interpolation) = $1.1 * 386 \text{ in/s}^2 = 424.6 \text{ in/s}^2$
- Response with 5% damping (Sa) – 1 G

Knowing that the spectral acceleration (S_a) is 1.1 G's or 424.6 in/s² (the response spectrum acceleration at the normal mode frequency), we can find the spectral velocity (maximum velocity at end of the beam) and the spectral displacement (the maximum displacement at the end of the beam).

$$S_v = \frac{S_a}{\omega_n} = \frac{424.6 \frac{in}{s^2}}{32.38 \frac{1}{s}} = 13.11 \frac{in}{s}$$

$$S_d = \frac{S_a}{\omega_n^2} = \frac{424.6 \frac{in}{s^2}}{32.38^2 \frac{1}{s}} = 0.405 in$$

The dynamic force can be determined from the weight at the end of the beam and the maximum acceleration ($S_{a_{max}}$). Then the dynamic stresses can be found;

$$F = ma$$

$$F_{dyn_{max}} = m * S_a = 1.0 \frac{lbs * s^2}{in} * 424.6 \frac{in}{s^2} = 424.6 lbs$$

Finally the dynamic stress is given by;

$$\sigma = \frac{Mc}{I} = \frac{(424.6 * 6) * (\frac{0.25}{2})}{(\frac{2 * 0.25^3}{12})} = 122,285 psi$$

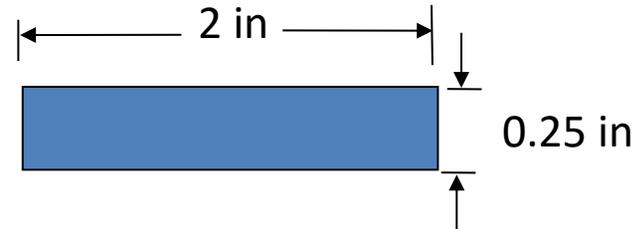
5.3 STEP 1: CREATING THE FE MODEL

Create a beam element with the cross section as shown. The length of the beam is 6 in.

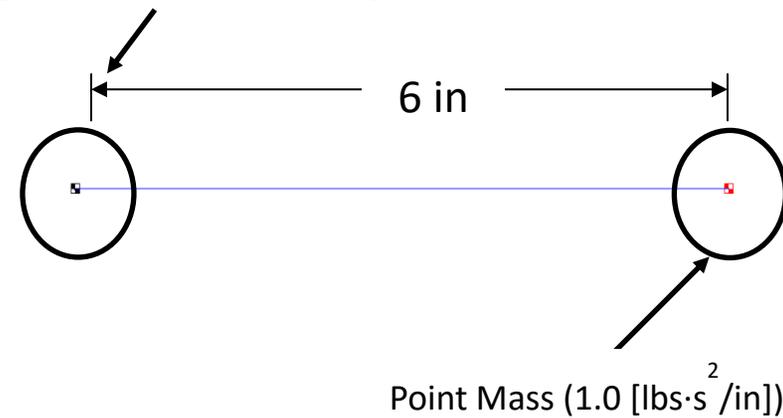
Add a point mass of 1.0 at one end of the beam. Add a large point mass of 10e6 to the end of the beam where the excitation will later be applied.

The cross section of the beam is chosen to be thin along the bending direction in order to minimize shear effects, this provides very close agreement between the FE results and the analytical results.

For this type of analysis, no load set is required, the analysis is performed by creating response spectrum, interpolation and damping functions. Then everything is tied together in the Analysis Manager as a Normal Modes/Eigenvalue analysis

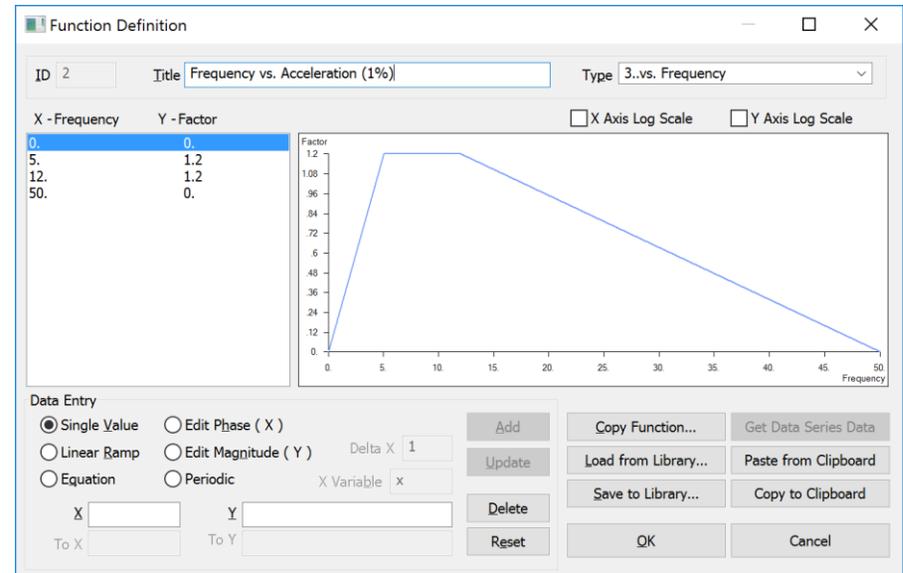
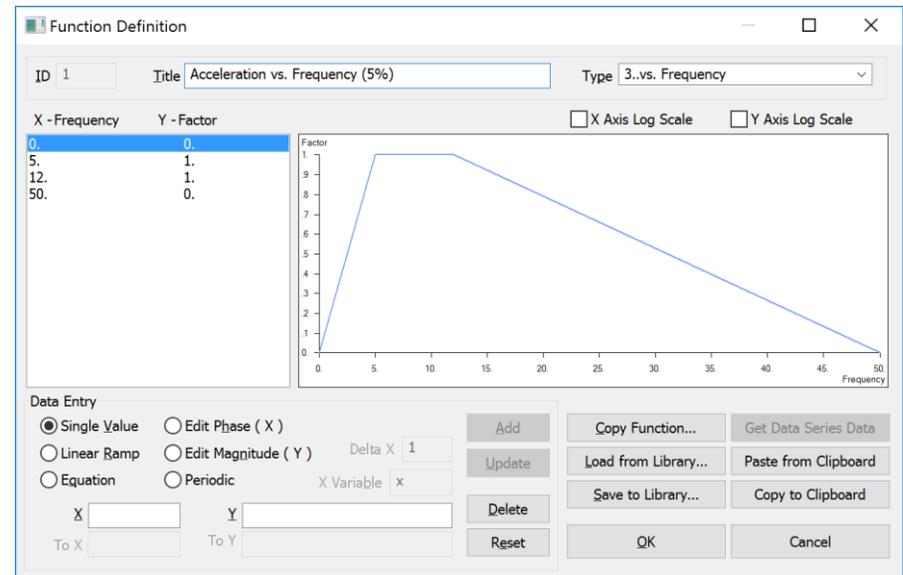
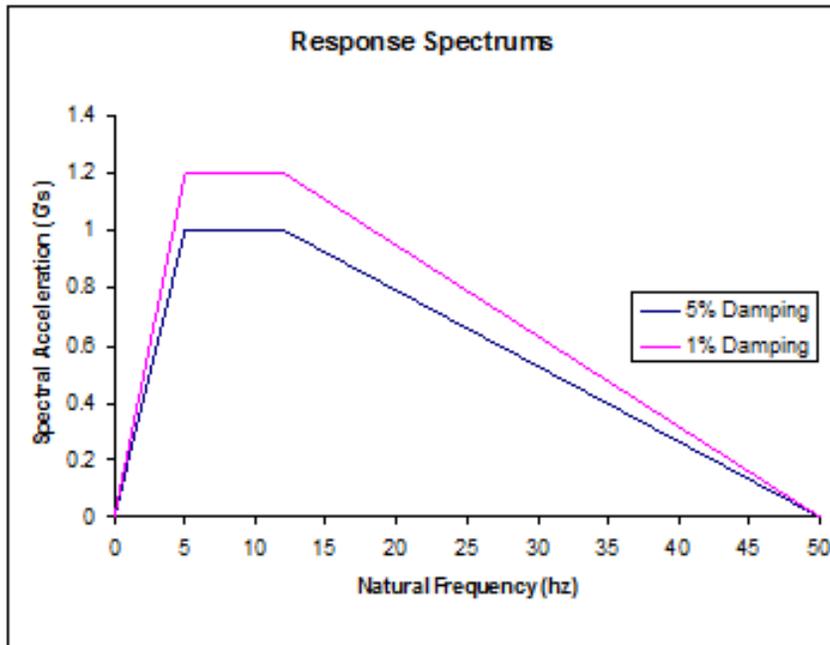


Large Mass for Seismic Analysis



5.4 STEP 2: DEFINING THE RESPONSE SPECTRUM

A total of 2 functions will be created to define the response spectrum for the analysis, they are Type 3..vs. Frequency. One will be created for the 5% damping curve (Function ID 1) and another for the 1% damping curve (Function ID 34). The curves are plotted below. Any number of response spectrums can be defined, depending on the requirements of the design. For this example, we will be using 3% of critical damping. The user can choose from Q damping, structural damping or critical damping



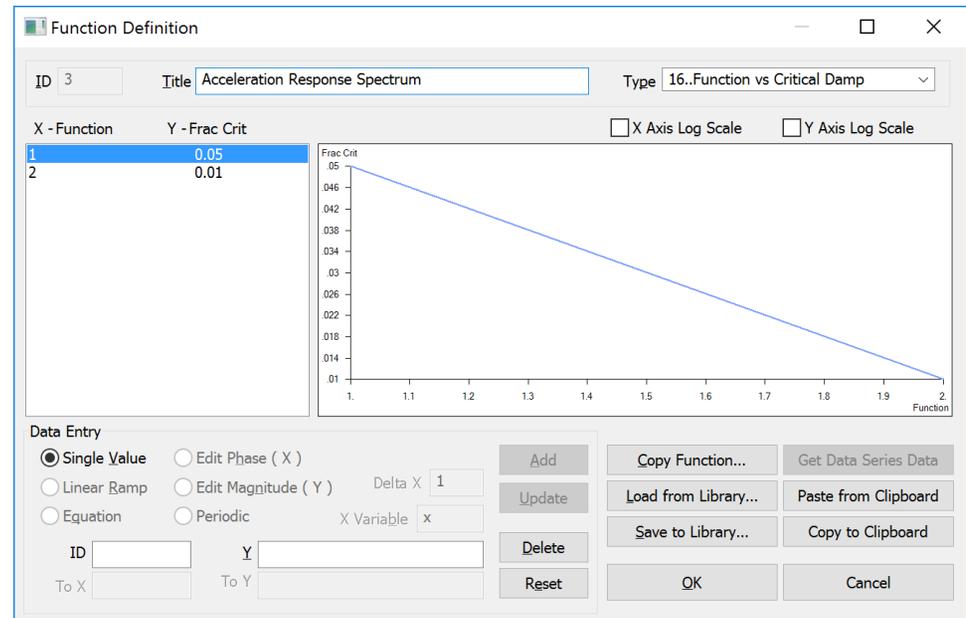
5.5 STEP 3: CREATING INTERPOLATION TABLE

Nastran requires an interpolation table to determine the system response, if the system damping is not given directly from a response spectrum curve. **IMPORTANT**, the interpolation table is mandatory, even if no interpolation is required between curves.

A function is created that assigns damping values to the previously created functions.

The function is Type 16..Function vs. Critical Damp. We will give the function a title of “Acceleration Response Spectrum”

A value of 0.05 (5% of critical damping) is assigned to function ID 1 and a value of 0.01 (1% of critical damping) is assigned to function 2. In this manner, the system response to a damping value other than the damping associated to the defined response spectrum curves can be calculated via interpolation between the defined curves.

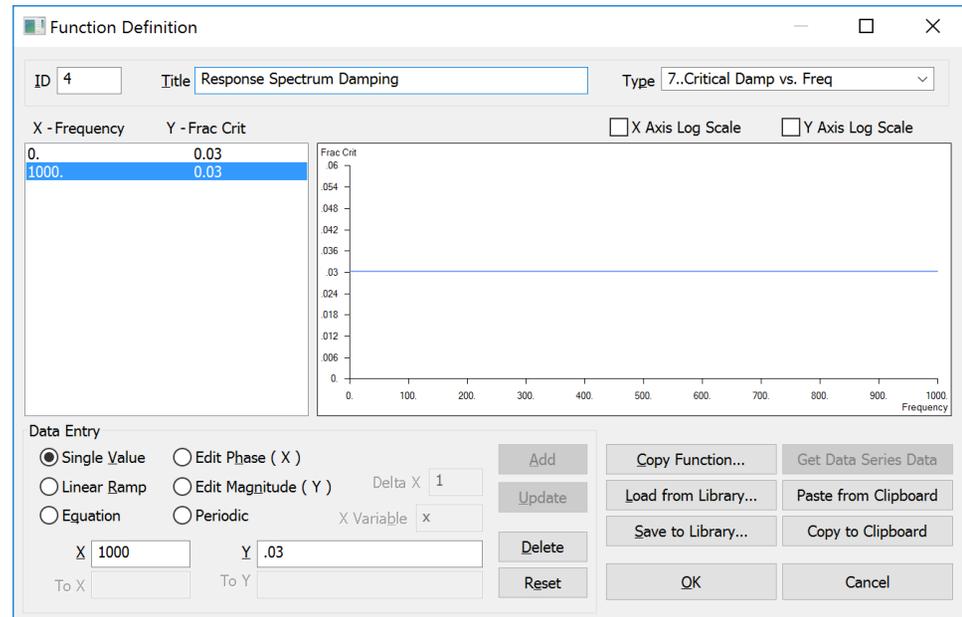


5.6 STEP 4: CREATING A MODAL DAMPING FUNCTION

The system damping can be defined as a function of frequency; we will use a 3% of critical damping across the frequency range.

The damping function is defined as Type 7..Critical Damp vs. Freq. We will give the function a title of “Damping”

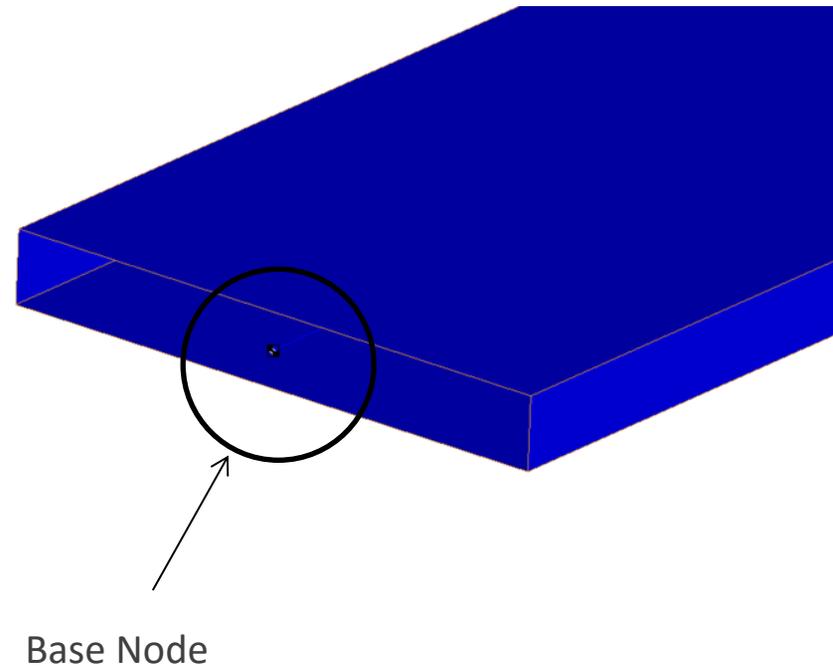
The function is used in association with the interpolation table. Since we have chosen a modal damping value of 3%, the system response will be found in the response spectrum between the 5% damping and the 1% damping curves. Note that if the system response at 5% damping was desired, the interpolation function previously created would only need to be defined for the function ID 1, and the damping function would then be defined as 5% over the frequency range.



5.7 STEP 5: CREATING THE EXCITATION NODE

For the response spectrum analysis, a foundation mass must be added to the excitation node. The mass should be 10^3 to 10^6 times the mass of the system.

Since this is a base excitation problem, and the base of the structure consists of one node, it is that node to which we will apply our acceleration response spectrum. In the case where the base of the structure is not one node, a rigid link approach is used to tie the multiple nodes of the base to a single node.

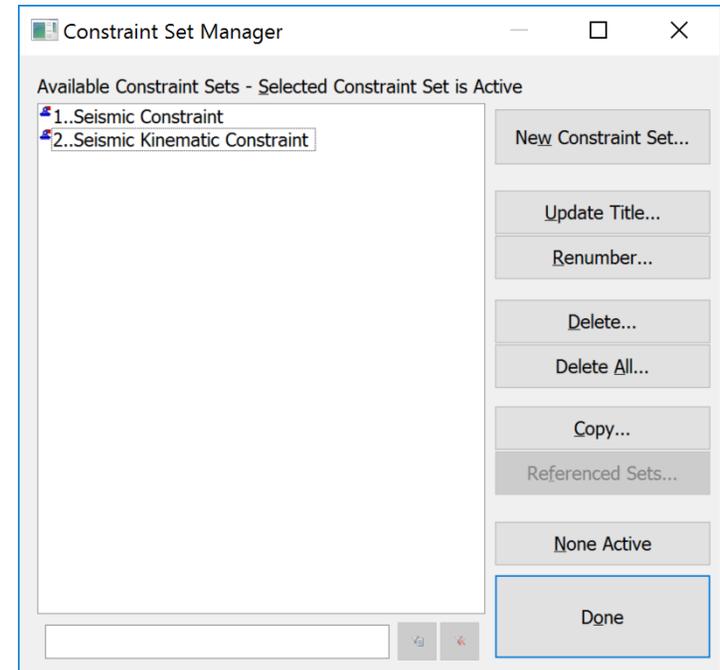
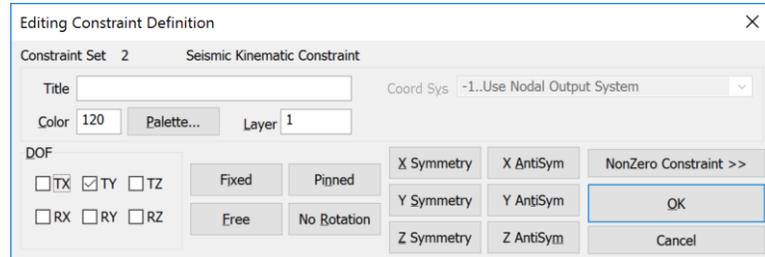
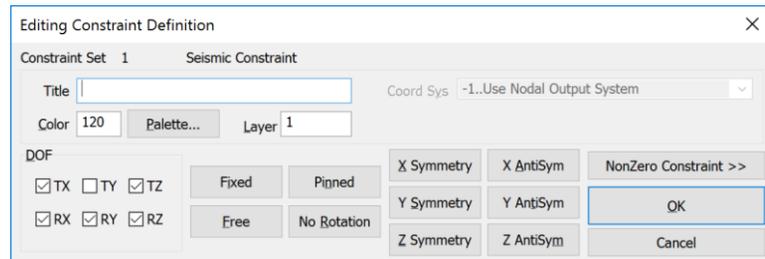


5.8 STEP 6: CONSTRAINING THE MODEL

In order to perform the response spectrum analysis, 2 constraint sets must be created.

The first set will constrain the model in all degrees of freedom except the excitation direction (in this example, y-direction).

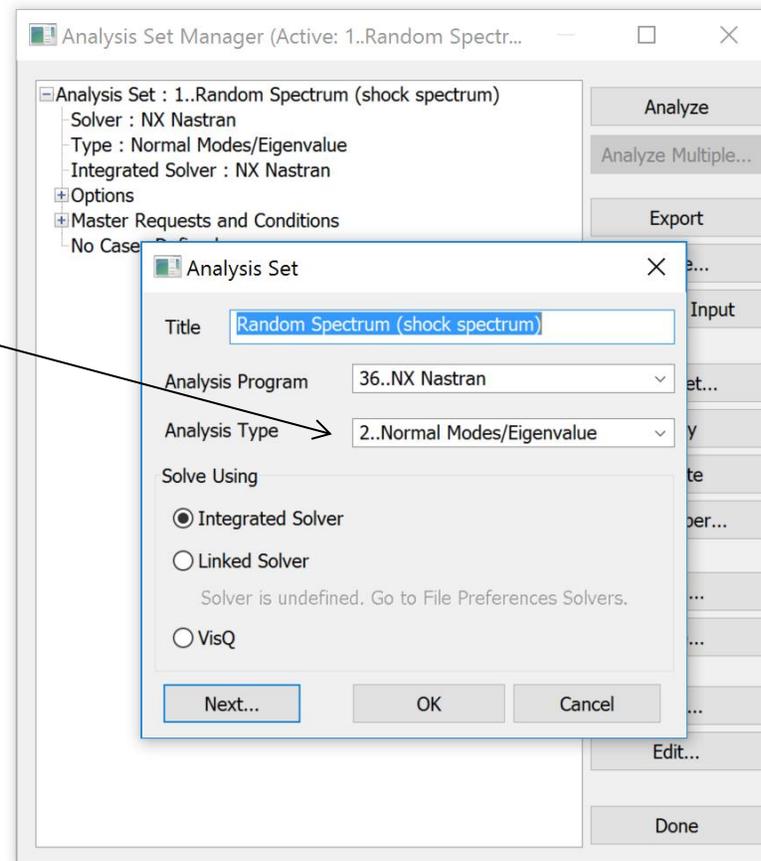
The second constraint set will be used as a kinematics DOF. This is known as a SUPPORT set in Nastran. The DOF that the excitation is in is constrained.



Note: Foundation mass not shown

5.9 STEP 7: SETTING UP THE ANALYSIS

Set up a new analysis set as type 2..Normal Modes/Eigenvalue.



Click Next button 5 times, in the Range of Interest, type in;

- 1.0 in the From (Hz) field

- 50.0 in the To (Hz) field.

This is very important since the first mode that will be calculated will be the rigid body motion (natural frequency of zero). If this field is left blank, the analysis will be performed from 0 Hz, this will give the incorrect value for the nodal velocity and displacement, and as such the stresses will be incorrect.

Note: The minimum frequency does not have to be 1.0, it can be any small number such that mechanisms are excluded.

NASTRAN Modal Analysis

Skip EIGx Method ID

Real Solution Methods

- Lanczos
- Auto (HOU/MHOU)
- Subspace

Legacy Real Solution Methods

- Givens
- Modified Givens
- Inverse Power
- Inverse Power/Sturm
- Householder
- Modified Householder

Complex Solution Methods

- Hessenberg
- Complex Inverse Power
- Complex Lanczos

Solution Type

- Direct
- Modal

Range of Interest

	Real	Imaginary
From (Hz)	<input type="text" value="1.0"/>	<input type="text" value="0."/>
To (Hz)	<input type="text" value="50.0"/>	<input type="text" value="0."/>

Eigenvalues and Eigenvectors

Number Estimated

Number Desired

Normalization Method

- Mass Node ID
- Max DOF
- Point

Mass

- Default
- Lumped
- Coupled

Complex Solution Options

Convergence

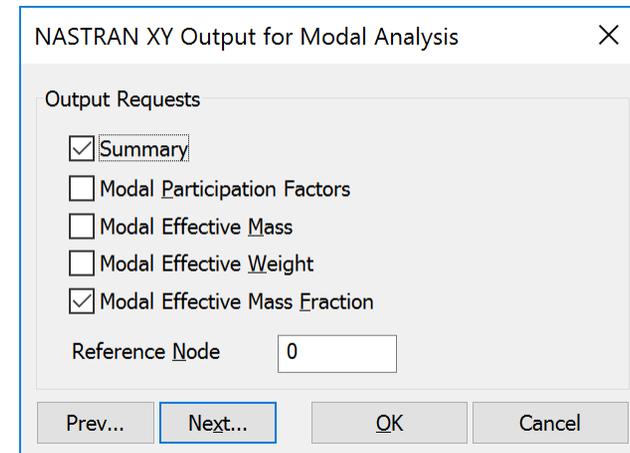
Region Width

Overall Damping (G)

Prev... **Next...** OK Cancel

Press Next button once. For the modal analysis, it is very useful to ask Nastran to output the Modal Participation Factors (PF). Select Summary and Modal Effective Mass Fraction. Nastran will output the PF's to the .f06 file. This is a very important tool when analyzing larger models that have multiple DOF and many eigenvalues. The PF's are used in combining the response from the individual modes based on the combination method selected.

However, you don't need to request'em since they are automatically calculated. This toggle just dumps them out to the f06 file and creates functions in Femap.



Press Next button once. This is the step that ties all of the functions together.

In the Spectrum – Spectrum Function ID drop down box, select “Acceleration Response Spectrum”.

In the Scale Factor box, type in 386, this will scale the response spectrum curve from G’s to actual acceleration values in in/s^2 .

In the Modal Combination - Method box, select any type since only one mode is calculated in the 1.0 Hz to 50 Hz range. Here we are using ABS method, but for this example, the same results are obtained with all methods.

In the Base DOF - SUPORT Set drop down box, select “Seismic Kinematic Constraint”.

In the Modal Damping – Damping Func drop down box select “Damping”

The screenshot shows the 'NASTRAN Response Spectrum Application' dialog box with the following settings:

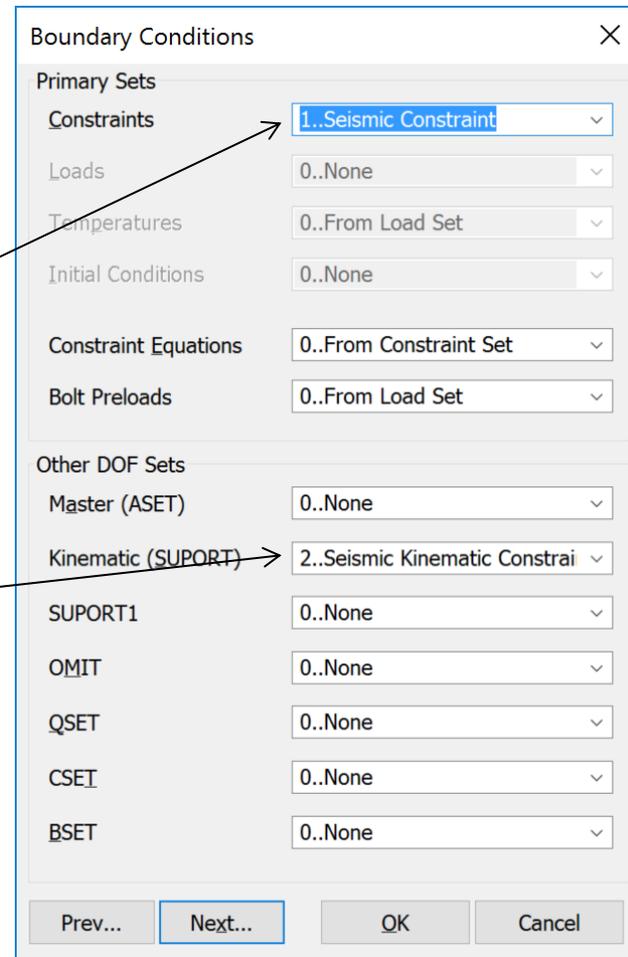
- Spectrum:**
 - None (radio button)
 - Acceleration (radio button, selected)
 - Velocity (radio button)
 - Displacement (radio button)
 - Spectrum Function ID: 3..Acceleration Response Speri
 - Scale Factor: 386.
- Modal Combination:**
 - Method: 0..ABS
 - Closeness: 1.0
- Base DOF:**
 - SUPORT Set: 2..Seismic Kinematic Constraint
- Modal Damping:**
 - Damping Func: 4..Response Spectrum Dampii

At the bottom, there are four buttons: 'Prev...', 'Next...' (highlighted with a blue border), 'OK', and 'Cancel'.

Press Next button twice.

In the Boundary Conditions – Primary Sets – Constraints drop down box select “Seismic Constraint”.

In the Other DOF Sets – Kinematic (SUPPORT) drop down box select “Seismic Kinematic Constraint”.



Press Next button once.

In the Nastran Output Request – Nodal select;

Displacement
 Constraint Force
 Velocity
 Acceleration

In the Nastran Output Request – Elemental select;

Stress

In the Customization area;

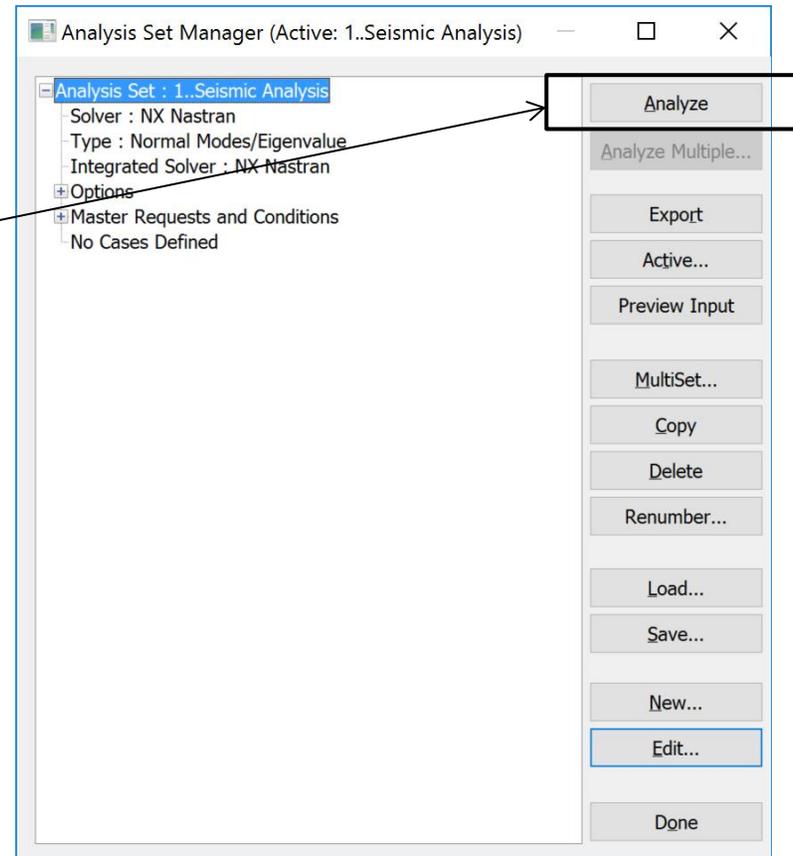
Set Results Destination to “2..PostProcess Only”

Click OK

The screenshot shows the 'NASTRAN Output Requests' dialog box. It is divided into three main sections: Nodal, Elemental, and Customization. In the Nodal section, the following items are checked: Displacement, Constraint Force, Velocity, and Acceleration. In the Elemental section, Stress is checked. In the Customization section, 'Element Corner Results' is checked, and the 'Results Destination' dropdown is set to '2..PostProcess Only'. The 'OK' button is highlighted in blue. Arrows from the text on the left point to these specific settings in the dialog.

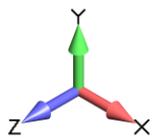
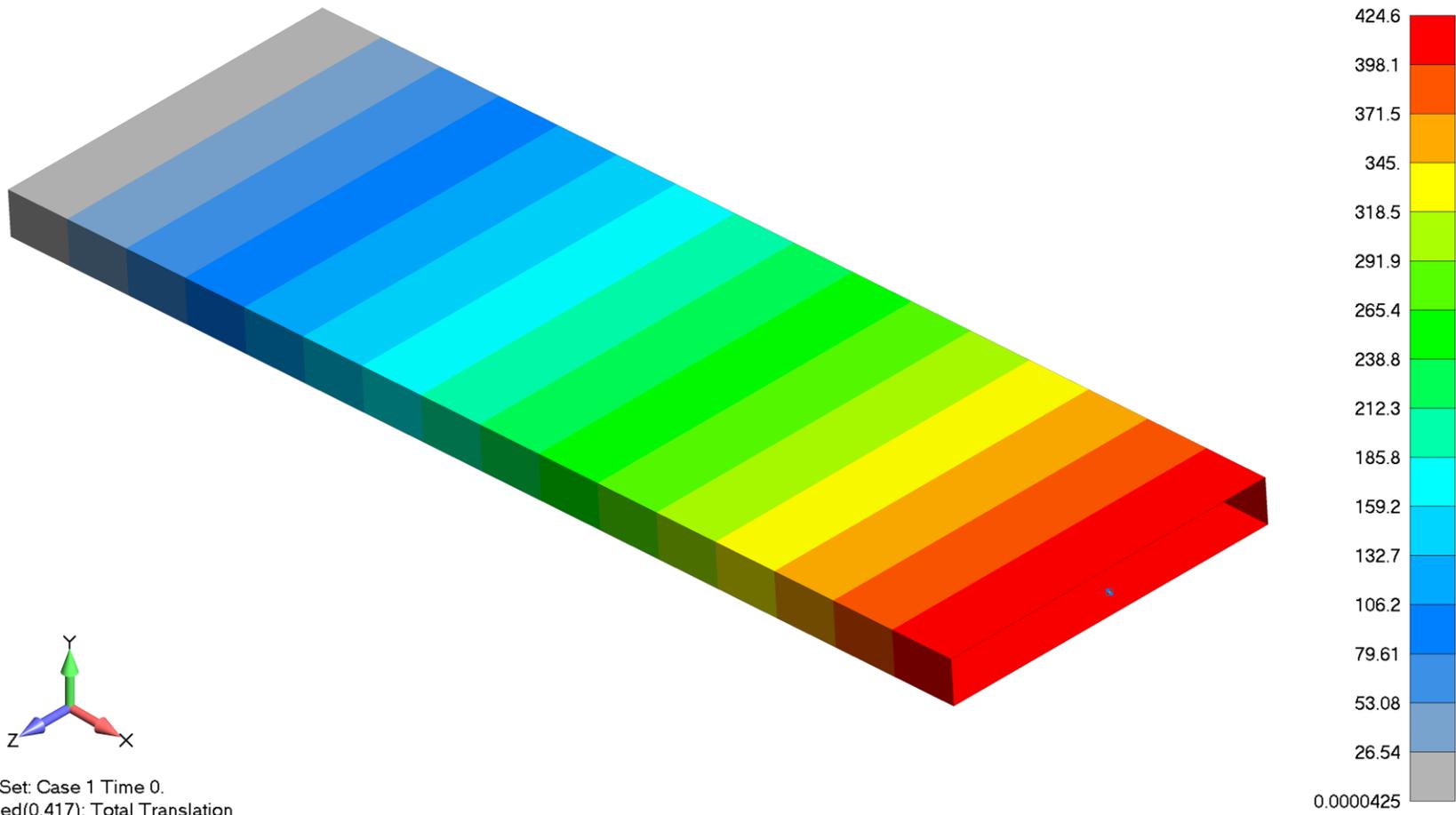
In the Analysis Set Manager, make sure that the Seismic analysis set is active, then click Analyze

We will also set up a Normal Modes analysis to determine the natural frequencies of the model (should get two eigenvalues, one being zero and the other being a bending mode).



5.10 POST PROCESSING THE RESULTS

Press F5 and select Beam Diagram from the Contour Style. Use the Post Data tool to look at the beam diagrams for the acceleration, velocity, stress etc. Below is the acceleration contour plot, note that the max acceleration is at the free end and has a value of 424.6 in/s².



Output Set: Case 1 Time 0.
 Deformed(0.417): Total Translation
 Elemental Contour: T2 Acceleration

Result Comparison

The results from the FE analysis follow very closely with the analytical results. The acceleration, velocity and displacement are exact (slight difference between analytical and FE velocity). Note that the natural frequency found in Nastran is slightly different than the analytical calculated value, this is because Nastran calculates the shear area of the beam. One can obtain the exact same results for the natural frequency in the FE model by excluding the shear area from the calculations, when this is done, the FE results match with the analytical results to 3 decimal places.

Results Comparison		
	Analytical	Nastran
Natural Frequency	5.1545	5.1509
Acceleration (in/s ²)	424.6	424.6
Velocity (in/s)	13.11	13.12
Displacement (in)	0.405	0.405
Stress (psi)	122,285	122,293

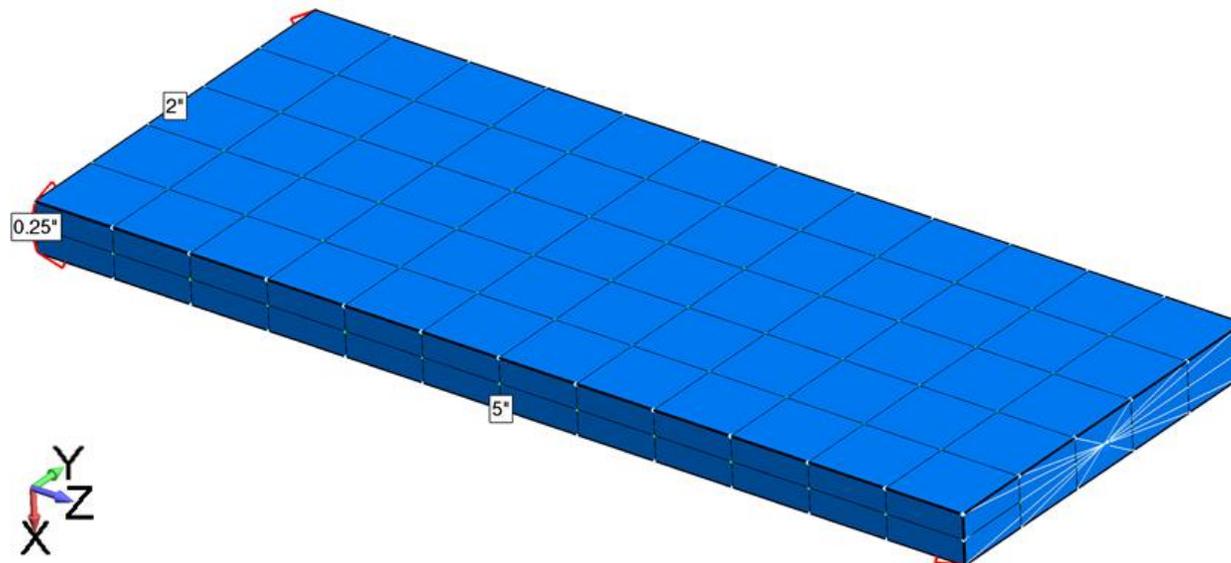
Extra Credit: Solid Meshed Beam

We can verify this method using a hex meshed solid beam. The beam is massless, with a point mass of 0.5lbf (1.30e-3 slugs) on the end. The beam properties are shown below:

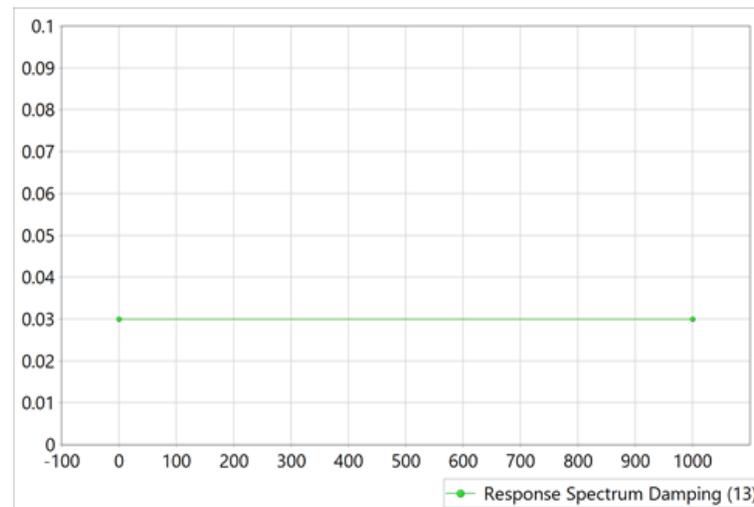
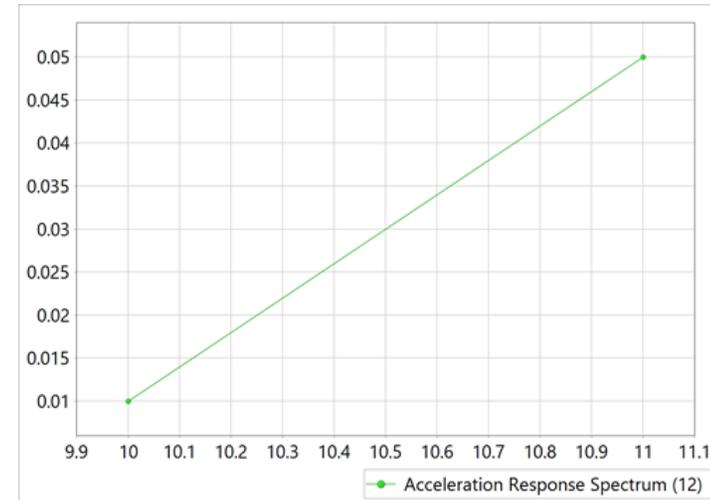
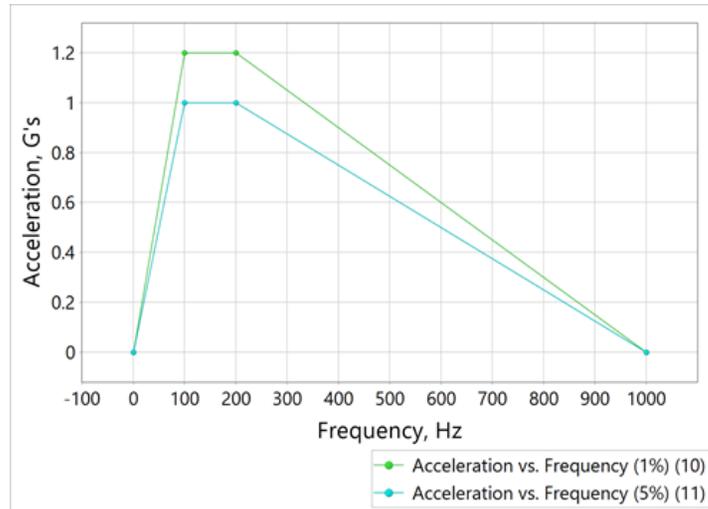
$$w = 2 \text{ in} \quad W = 0.5 \text{ lbf}$$

$$T = 0.25 \text{ in} \quad E = 10e6 \text{ psi}$$

$$L_{\text{beam}} = 5 \text{ in} \quad I_{xx} = 2.6e-3 \text{ in}^4$$



An Acceleration vs. Frequency spectral plot was created for 1% and 5% damping, an interpolation function was created to find intermediate damping values, and a damping function was generated.



Let's first take a look at the hand calc's to show how the beam is expected to behave.

Deflection is calculated first:

$$Y_{max} = \frac{WL^3}{3EI_{xx}} = 8e - 4 \text{ in}$$

Based upon this end deflection, the beam's resonant frequency can be calculated as:

$$f_n = \sqrt{\frac{1}{2\pi} \left(\frac{g}{Y_{max}} \right)} = 110.6 \text{ Hz}$$

Spectral acceleration for this condition is

$$S_a = 1.1 * 386.09 = 424.6 \frac{\text{in}}{\text{s}^2}$$

Continuing, we can approximate the spectral velocity, displacement, and stress:

Velocity:

$$S_v = \frac{S_a}{\omega_n} = 0.611 \frac{in}{s}$$

Displacement:

$$S_d = \frac{S_a}{\omega_n^2} = 8.80 * 10^{-4} in$$

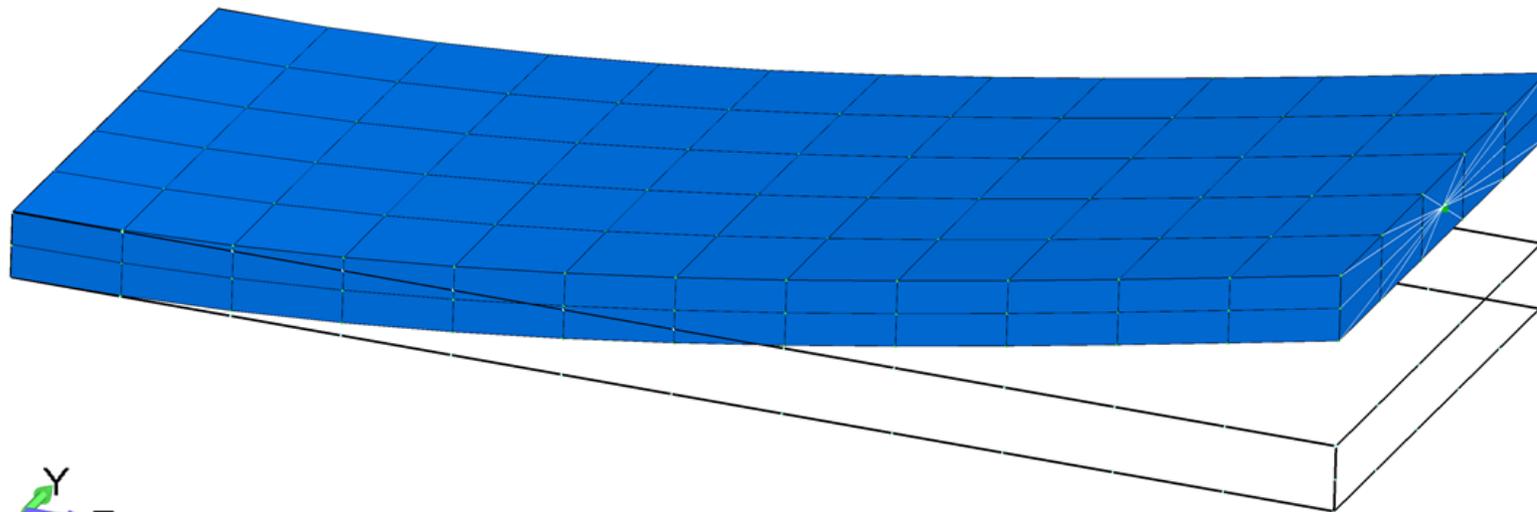
Stress:

$$F_d = W * S_a = 0.55 lbf$$

$$Stress = \frac{Mc}{I} = \frac{(F_d L) \left(\frac{T}{2}\right)}{I_{xx}} = 132 psi$$

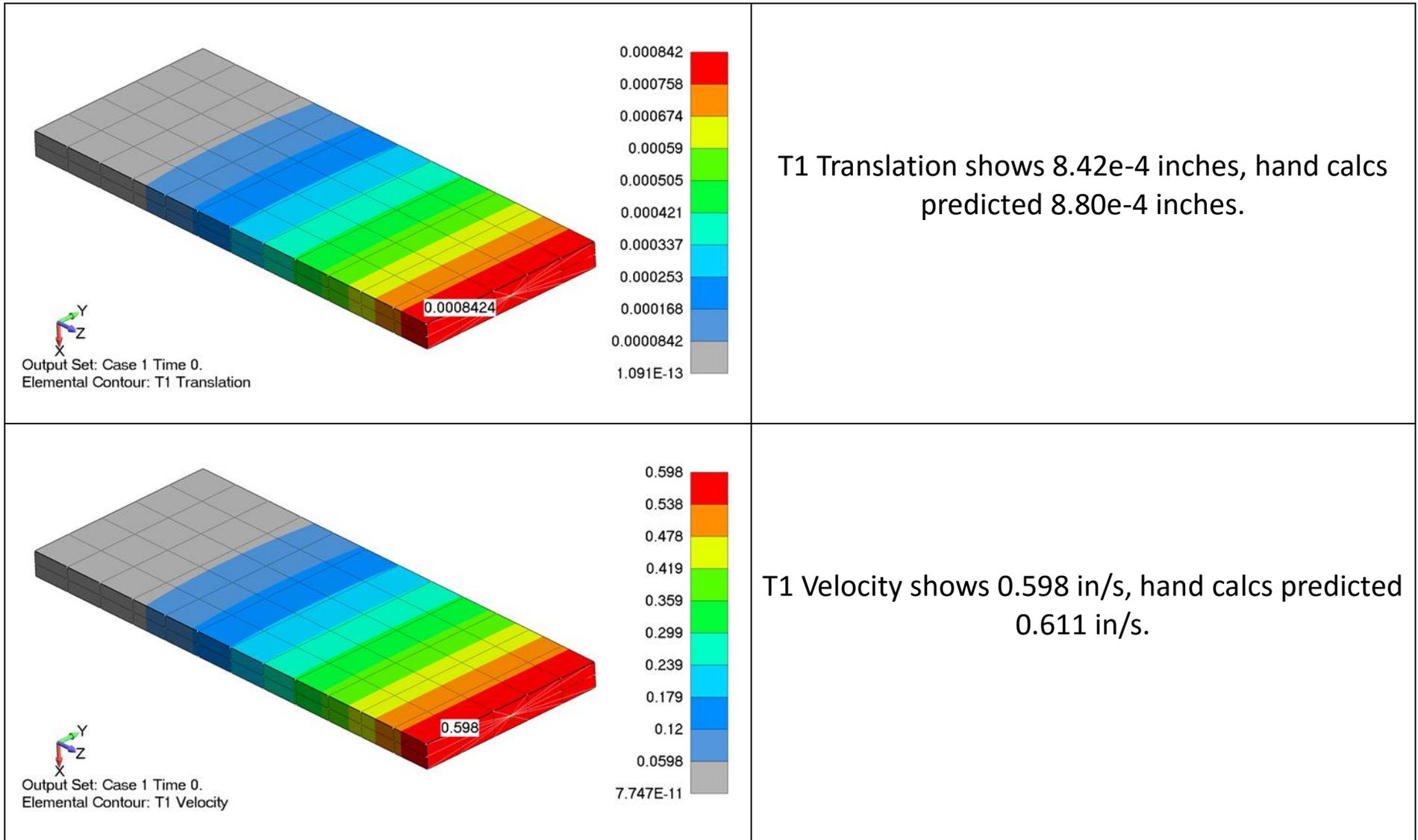
First up is a normal analysis for the solid model, the first mode is at 113 Hz:

- 2..NX Nastran Modes Analysis Set
 - 2..Mode 1, 112.9961 Hz
 - 3..Mode 2, 845.2501 Hz
 - 4..Mode 3, 4466.077 Hz

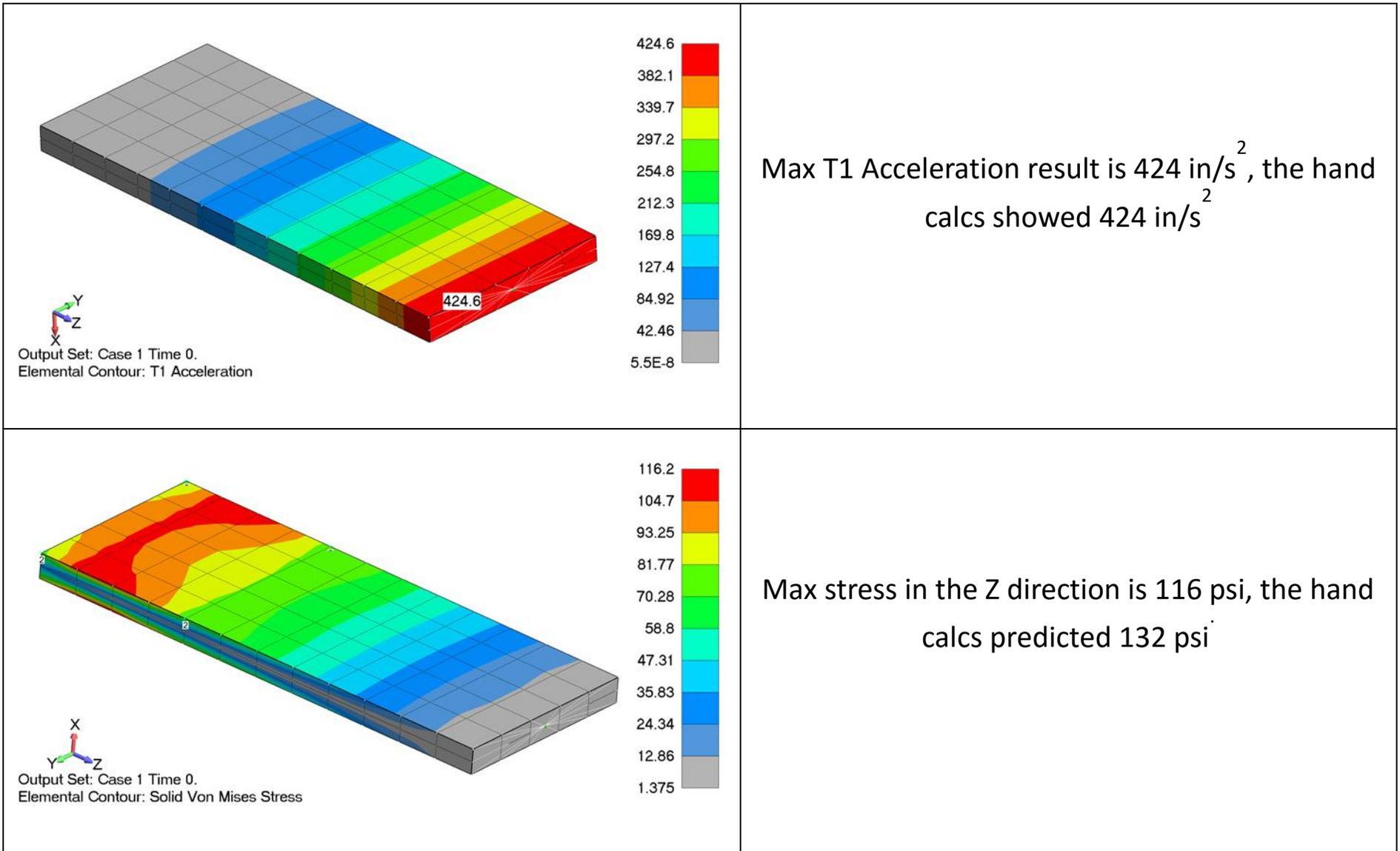


Output Set: Mode 1, 112.9961 Hz
Deformed(27.79): T1 Translation

With the normal modes calculated, next is a shock response analysis:



With the normal modes calculated, next is a random response analysis:

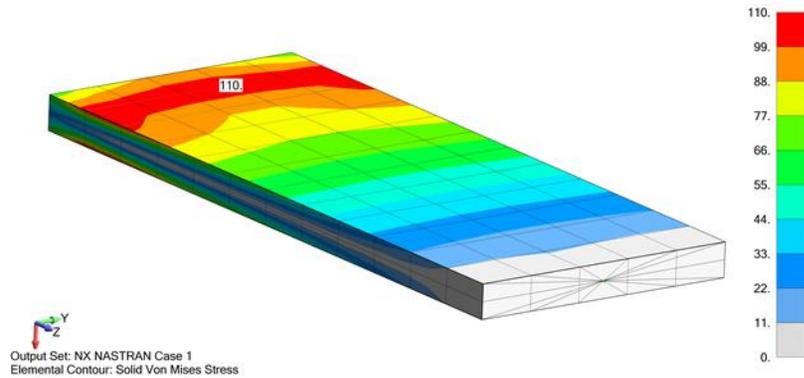


The FEA results for the solid model are not as close to the hand calculations as the beam element, but you can see the relative accuracy below:

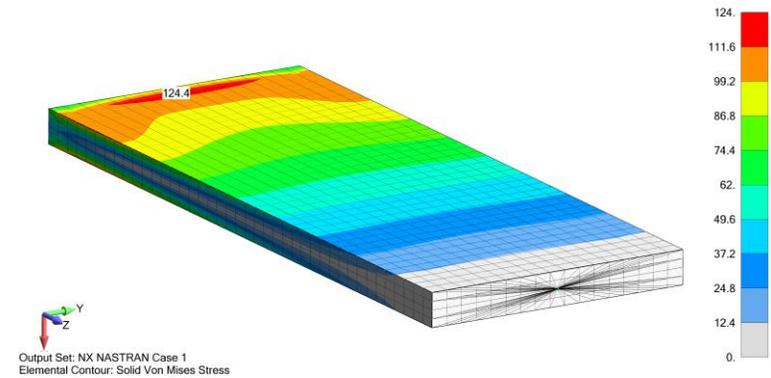
Results Comparison		
	Analytical	Nastran
Nautral Frequency (Hz)	110.6	113.0
Acceleration (in/s ²)	425	425
Velocity (in/s)	0.611	0.598
Displacement (in)	8.42e-4	8.80e-4
Stress (psi)	136	116

The error in the maximum stress of the FEA model is 12%, which seems a bit on the high side. Let's check a hand calculation of a 1G static load versus the finite element static load to explore the behavior. This beam with 1G static loading should see 120psi maximum stress based on the simple calculations.

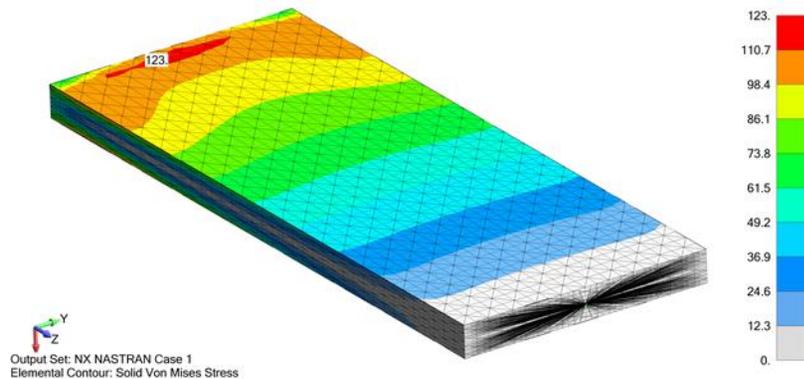
Maximum stress is 110 psi with a hex meshed solid element beam, which is 8% error from the hand calculations. This is similar to the error seen in the dynamic analysis.



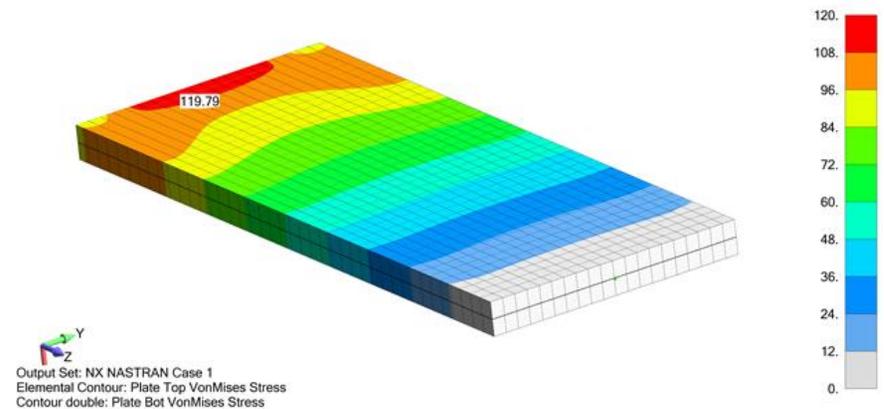
Maximum stress is 124 psi in a fine hex meshed solid element beam. This gives an error of just over 3% compared to the hand calculations.



Maximum stress is 123 psi with a 10-node tet meshed solid element beam. This gives an error of under 3% which is a large improvement.



Maximum stress is 119.8 psi with a quad meshed plate element beam. This matches with the hand calculations very closely with an error of 0.2%.



It is interesting to compare the accuracy with the # of nodes required for analysis. In a solid element model it seems that the coarse hex mesh does not have enough definition to capture the stresses and has a relatively large error. The fine hex mesh drops the error down to 3% with just under 4,000 nodes. Meshing with a fine 10-node tetrahedral mesh gives similar accuracy but requires 26,000 nodes, a significant increase! The plate meshed beam is the closest with 0.2% error and under 900 nodes.

A summary of the findings for a 1 G static load is shown in the table below.

	Max Stress	% Difference	# of Nodes
Hand Calculation	120 psi	n/a	n/a
Coarse Hex Mesh	110	8%	236
Fine Hex Mesh	124	3%	3,782
Fine 10-node Tet Mesh	123	3%	26,274
Fine Quad Plate Mesh	119.8	0.2%	884

Conclusions

Did you like it? If anything, writing this tutorial was a great tool for us in developing a core understanding of response spectrum analysis. It is a start into the world of advanced dynamics and we would welcome your questions and comments about this tutorial.